

# DESIGN AUTOMATION OF CAM LOBE MODELING IN CREO USING C#

Harish Reddy M<sup>1</sup>, Dr. P. Srinivasa Rao<sup>2</sup>

<sup>1</sup>Student of M.Tech CAD/CAM, IARE, Telangana, India

<sup>2</sup>Professor, Dept. of Mechanical Engineering, IARE, Telangana, India

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**Abstract** - In mechanical engineering, the procedure of design requires additional time in view of its iterative and proficient processes. For design, the availability of CAD packages in market, are so many which requires modeling skills. Many efforts had been made in the area of design automation like integration of CAD software with different computer programming languages. In between these CAD software and computer programming language, intermediate software is responsible for the interface between them.

In this study, the Modeling of cam lobes has been done to validate the idea of Design Automation. Cam lobes are responsible for opening and closing of valves in an engine cylinder, which are appeared on the camshaft. Cam lobes modeling process using CAD software like Creo involves an integrated approach which is tedious. It will take minimum of 15-20 minutes to model a cam lobe. If there are more lobes in an engine (say 20), then modeling all these lobes will take at least 3-4 hours and modeler should be very much familiar with CAD software. The objective of this project is to develop an application using C# programming language to automate cam lobe model. Using this application, we can design 'n' number of lobes in less than five minutes.

**Key Words:** Design Automation, Creo, GUI, Parametric modelling, C#

## 1. INTRODUCTION

### 1.1 Knowledge Based Engineering

It is closely related to the domain of manufacturing design and production. In KBE, the use of Knowledge based technology is to support CAD and the knowledge based techniques can be applied to the entire product life cycle. KBE systems provide more flexibility in the development of design automation. According to W.J.C. Verhagen et al. [1], KBE is a life cycle with various stages like identify, justify, capture, formalise, package and activate the design. In the process of design, use of KBE framework we can save time and which makes a predominant role in the market [2]. It is quoted that in CAD without KBE, the creative design is of 20% and routine design is of 80%. Whereas with KBE, an additional time profit in the design process occurs. The conceptual benefit of KBE is clearly shown in the below figure and also mentioned about the more time focused on creativity [3]. The ultimate goal of KBE is to save the time and reduce the cost of design through automation.

### 1.2 Design Automation

In research field many efforts have been made in the area of design automation. According to Christian Johansson et al. [2], "Design automation deals with automating design tasks which are normally carried out in manual fashion by engineers and designers". In simple, Design Automation creates shortcuts to quickly accomplish common design tasks. In order to understand design automation we have to go through with Knowledge Based Engineering. Benefits of KBE include the automation of parts of a product life cycle. With use of DA, we can develop a Graphical User Interface (GUI) or tailor-made software in design process with collaboration of computer programming language and CAD software. IndrajitSinh J. Jadeja et al. [4] discussed about the methodology used having three phases; they are design phase, integration phase and Modelling phase. Till date we have observed only the Modelling phase but not the analysis phase using GUI.

### 1.3 Parametric Modeling

In CAD/CAE, parametric Modeling plays a predominant role as it is used to change the shape of model geometry as soon as the dimension value is modified. The important feature of parametric Modeling is to automatically change the attributes that are interlinked with the model or component. The conceptual model of the research approach with parametric design is discussed by Marius Marian Cucos et al. [5]. These type of parametric Modeling is generally seen in advanced CAD software like CATIA, SOLIDWORKS, CREO, etc. which saves Modeling time.

## 2. LITERATURE SURVEY

IndrajitSinh J. Jadeja et al. [4] discussed about the research work carried out on Flange Coupling using GUI based design software. Considered a step by step methodology i.e.,

- Design Phase
- Integration Phase
- Modelling Phase

In design phase, selection of programming language and a component took place along with design variable and programme structure. Integration phase uses export values from design phase and used to import the values to modelling phase. In this work [4], the programming language used in Design phase is Visual Basic with input variables of Flange coupling, Integration phase by MS Excel and the CAD software used in Modelling Phase is CREO. With input variables in Visual Basic, the output variables with design equations are obtained and these variables are exported to MS Excel sheet. The solid model created in CREO is integrated with MS Excel sheet to regenerate the model and updated dimensions are obtained as per the sheet values. As this work reduces the design time and increases the production efficiency with this type of GUI or Tailor-made software.

Ramakrishnan BalajiIyer et al. [6] discussed about the design optimization of CAD model using Creo simulation result and computation in MathCad as an integrated approach. In this work researcher links the Creo simulation result as the input in the MathCad and output of this program is again directed to Creo Parametric, then using this output result CAD model is optimized. The step by step procedure carried in this work, primarily the design of the model in Creo and defining the design parameter which is to be optimized. Then the analysis of the model is carried in the Creo Simulation and these results are integrated with the MathCad to optimize the design, which is later validated with theoretical values.

The research work by Chanwoo Park et al. [7], discussed about the design optimization of an Aircraft Wing using Multidisciplinary Design Optimization (MDO) via, Parametric Modeling. A fully automatic computational capability in a MDO system was developed and parametric Modeling was used for analysis. MDO framework structure made up of two groups. The first one is an automated analysis, consisting of an interface module, a parametric Modeling module, an aerodynamic analysis module, a load transfer module and a structure analysis module. And the second one is an optimization group consisting of a response surface module and an optimization module. Dhaval B. Shah [8] discussed about the Flange Coupling model created in Autodesk Inventor which was integrated with Excel spreadsheet to reduce the cycle time of design modeling. The dimensions required for modeling was calculated in Excel sheet using empirical formulas and exported to Inventor design software. Initial input data are physically entered in Excel sheet and later output data are obtained. The 3D model of Flange coupling was created in Inventor and using the parameter feature, model was dimensionally allotted with specific characters. Then the Inventor software is linked with Excel sheet using parameter feature. After successful linking of Excel sheet, the model in Inventor is regenerated with the obtained values from the sheet.

Yogesh H.Sawant et al. [9] discussed about the Design automation of a Screw Conveyor Components in CATIA using Visual Basic for Application (VBA) and Excel sheet. In this work, the dimensional calculations are done in Excel sheet using Loop function feature. The input parameters of Screw conveyor components was entered in the function and output dimensions data obtained automatically, which was later integrated with CATIA. VBA which was inbuilt in CATIA, helped in generation of components automatically. The screw conveyor model having different components and sub-components are automatically generated by VBA code according to input parameters. The work concludes the scope of programming language in design and time in modeling a component is reduced.

Umesh A. Bedse et al. [10] developed Graphical User Interface (GUI) based design software using Visual Basic programming language integrated with Creo CAD software for the design and Modeling of CI engine. Discussed about the Design Automation by developing GUI in Visual Basic and then output data was transferred to Excel sheet. After that, integration of Excel sheet with Creo took place for simulation of CI engine model. Initially in Visual Basic, the CI engine components are coded to calculate the dimensional values of each component with some input data. The output dimensional values obtained in GUI was transferred to Excel sheet and later it was integrated with Creo software to regenerate the model with latest dimensional values of CI engine components.

### 3. EXPERIMENTAL STUDY

#### 3.1. Conventional modeling in Creo 2.0

The process involves an integrated approach which seems lengthy. This process takes time if one has to modify previously modeled camshafts as this process takes into account the feature based Modeling practice from very beginning in order to make it easy for future changes.

To demonstrate the process, I assume that other features (ex: journal, dowel hole and key slot etc.) are already created before lobes. That gives us a reference for our lobes modeling process. It is very important to keep standard reference planes and axis as our reference when we need to create subsequent feature like lobes.



Fig-1: Camshaft model without Lobe

##### 3.1.1. Creation of a Standard View

First step is to create a view which is standard for all lobes from where we can see lobes oriented at different angles with respect to single point of reference. This single reference can be either dowel hole or key slot. For instance, give this view a name as "LOBE\_ORIENTATION".

It is very important to pay enough attention while creating this view as we are going to use this view in process of creating all lobes and drawing items.

Key Points for Creating LOBE\_ORIENTATION view:

- It should face to front of the screen at the end from where we see all orientation angles.
- As shown above, LOBE\_ORIENTATION view must adjust such a way that dowel hole comes at the top. The reason behind this is, we always measure lobe orientation angles with respect to dowel hole in counter clock direction.

##### 3.1.2. Points File Data to Creo

This is a compilation of data generated from Dynasty (Dynamic Analysis Simulation Software). ".pts" file can be opened using WordPad. We are going to use these data in our Creo model at the first place after creating a standard view.

Process of Importing Points Data

Insert--->Model Datum---> Graph

We have to give a standard name which defines its significance. Here, we consider each lobe with unique number generated from CamDesign software. Now, prefer to use that number as graph name. In this reference model we have imported 4 different points file for creating different lobes. They are 783,784,785 and 786lobes. Once we name it as per lobe number, a sketch screen opens with a grid. We have to import our data to this sketch screen as a graph with specific details as discussed below.

- Create a coordinate axis for graph
- Creating lines that represents 360 degree
- Creation of Lobe Profile Points in the Graph

### 3.1.3. Creation of Lobe

#### 3.1.3.1 Creating Plane for Lobe Start Location

For First Lobe - Inlet Number 10 - the distance from datum was 67.3 mm. Name this plane such a way that, it can signify its type, number and its purpose. As shown below, it is named as "IN10\_DISTANCE" which describes that it is inlet lobe for cylinder 10 and the plane itself is the starting point of the lobe surface, distance measured from datum.

#### 3.1.3.2 Creating Plane for Lobe Orientation

- Bring your view on the screen to standard "LOBE\_ORIENTATION" view which can show the dowel hole in top position and a plane passing through it. Always select center axis A that passes through the entire camshaft for reference whenever it is necessary to choose an axis as a reference from model tree.
- Select axis A as your first reference and "ORIENTATION\_DOWEL" plane as your second reference for creating your orientation plane for Inlet-10lobe.
- Enter the value of orientation angle required for this lobe. Recall the list of items required to make a lobe had this value. Then take the value from the following information table provided.

So now, we have created a plane passing from axis A and oriented at 312.49 degree counter clock direction with respect to "ORIENTATION\_DOWEL" plane. Such an orientation plane for each lobe signifies the maximum lift point of the lobe.

#### 3.1.3.3 Creating a sketch of trajectory of center of roller which rolls on the lobe profile

Here, we will first create the path of roller which rolls on camshaft lobe and then we will create an offset path by roller radius to actually create the lobe profile. In order to create this roller trajectory, we will have to create a sketch which is shown in next step. This process is common for all lobes creation.

- Select plane "IN10\_DISTANCE" as our sketching plane.
- Make sure that we have right sketch setup to create our sketch. For this we need to follow following steps: i) after selecting a sketch plane, see in which direction your sketch setup orients. If it does not become as our standard view "LOBE\_ORIENTATION" then bring it to that view by applying following settings.
- Select sketch references:
  - i) Axis A (Always)
  - ii) Orientation Plane for the lobe, in this case IN10\_ORIENTATION.
- Sketch a circle with  
Radius= (Cam Follower/Roller Radius + Base Circle radius)

We are creating a path of roller/cam follower center's trajectory. We will use our points file to describe its lift from nominal position. But in closed condition, the roller/cam follower follows on the circle with above diameter.

- Now recall the list of items discussed from excel sheet. For 783 lobe, these values are 27.5 and 33.125. So now we will sketch a circle with radius  $27.5+33.125 = 60.625$  (i.e., diameter 121.250) placing center at axis A reference point.
- Now, we actually do not need entire circle. We need to consider only part of it which falls between the angles of OPENING (-65) and CLOSING (+80). So let's trim the remaining lines by creating a boundary of these angles.

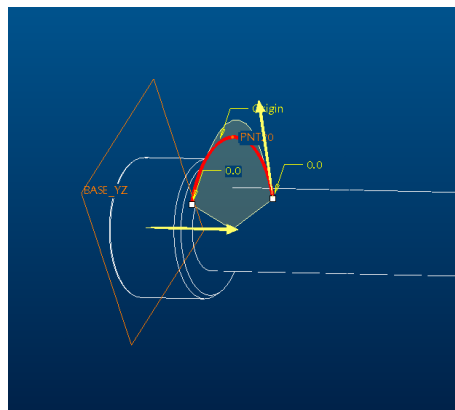
Now here, it is important to understand which side we put our opening angle and which side we put our closing angle. We know that orientation angle is 312.49 for Inlet-10 lobe and hence we can make sure here that it stays in 4th quadrant. So we will dimension our angles from that reference which is in 4th quadrant.

- After placing correct angles on correct side of reference plane (orientation plane), we will trim extra lines as shown below. After trimming extra lines, our sketch looks like shown below. Please note that the boundaries created by placing two lines from center axis are just construction lines and not part of the sketch.
- Now using this sketch our next step is to create the trajectory of roller center.
- Select the sketch and click on Variable Section Sweep tool.

The yellow arrow shows the starting point of our profile curve for roller trajectory (which is not at correct location by default). Now to be clear, profile curve (i.e., 783) in the model tree is read from -65 to +80. That means, it is always read from -ve to +ve direction of angles.

Now we will setup the relation to read all points from 783 graph to create the trajectory.

- 1) Dimension the distance between PTN20 and the point beyond PTN 20 as 107.1
- 2) Go to Tools--> Relations and type following relation as shown below for the dimension just placed between PTN 20 and the point beyond PTN 20. There are 5 important things to understand about this relations shown below



**Fig-2:** Trajectory of roller or cam follower

The relation is:

```
sd4 = evalgraph("783", trajpar*145-65)
```

This has to be typed same as shown. The arguments inside this relation change, when we change the lobe.

1. sd4 is the dimension that we had placed between PTN20 and the point beyond PTN 20 and we are driving this dimension using the relation expressed after " = " to sign.
2. evalgraph is the function which will evaluate this function by reading the argument inside this function.
3. "783" is the graph name we had given from which all points data will be read starting from -65 to +80
4. trajpar - function decides to make trajectory using the extracted points from 783graph
5. Angle Value - 145: This is very important. This value is the total included angle between opening and closing. That means in our case it is  $65+80 = 145$ . Trajpar function acts for total of 145degree
6. angle value -65 This decides the start point for trajpar function to act so if we this argument as 65 then trajpar function will extract points from 783 graph starting at -65 to total of 145 degree in increasing direction.

Click on Variable Section Sweep in model tree (i.e., keep it selected) and again click on Variable Section Sweep tool. Select the trajectory of roller/cam follower if not selected already and do not change any set up on this one. Here we just need to create an offset path of the same trajectory by amount of roller radius in internal direction. After having this, click on sketch tool shown in

the same panel. It will orient in some orientation for sketch purpose. Do not bother about orientation here as we just need to create an offset path.

Draw the line starting from the point shown below and in the internal direction. Change the dimension to roller radius that is 27.5 mm taken from the excel sheet. It should create an offset profile. Please note that this offset profile itself is our lobe profile. Now you should have another section sweep in the model tree which is your lobe profile. The only step remaining now is to extrude this up to appropriate lobe width.

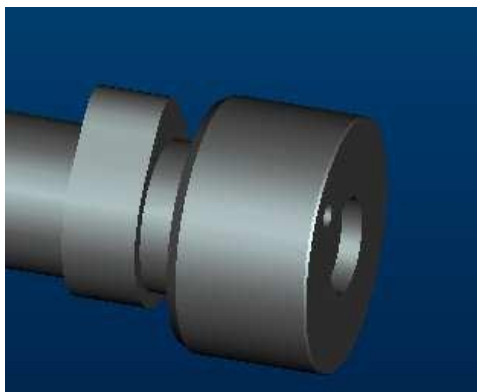
#### 3.1.3.4 Create an extrude using the offset profile just created.

1) Select a sketch plane (always choose the plane we created for lobe distance, here it is IN10\_DISTANCE) because that is the starting point of the lobe surface.

2) Select extrude tool -->go to placement and select Define

It will orient at some random orientation but we have to make sure that we have right orientation. Always choose your base standard view as your orientation ("LOBE\_ORIENTATION" view). And then Sketch by selecting sketch reference as Axis A and Select Use Tool from Tool bar. Then capture the offset trajectory created in previous process

And now, create an arc passing from center axis A and completing the trajectory. If you would not have selected the "tangency" relation in the 783 graph, you will never be able to complete this sketch by connecting with an arc like this. Then select an appropriate direction for extrusion. (Make it opposite here). And then enter the lobe width = 26.9 here.



**Fig-3:** Cam lobe model in Creo

## 3.2 Modeling with C# application

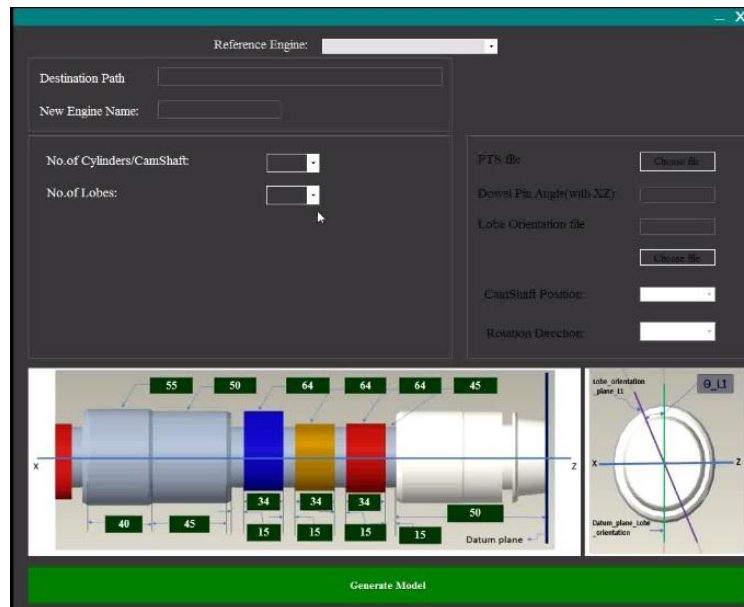
### 3.2.1 Design Phase

In order to develop the GUI for camshaft and cam lobe, we have to follow some program structure that the main window displays. It mainly comprises of input data that has to be given for the changes in the model. The program that has to be designed is written in programming language, C#. After successful execution of program, the window displaces with some buttons. The Design interface of the cam lobe model is created using input variables if lobe in programming language. The input variables that are considered for cam lobe are

- Opening angle of valve: at which the valve starts opening, at the angle of crankshaft rotation before the intake valve.
- Closing angle of valve: at which the valve comes to closing position, the angle of crankshaft rotation after the intake valve.
- Valve timing: the duration of valves in the opened and closed state, which can be determined from Valve time diagram.
- Valve Lift: the distance between the valves during fully opened and closed state.
- Engine RPM: the crankshaft rotational speed is considered for calculations of valve.

- Pitch curve: it defines the function of curve in uniform velocity, uniform acceleration, and simple harmonic motion of valve during the opening and closing state.

After successful input variables, the Graphical User Interface is developed as shown in the figure. The window has two input options initially, i.e., number of cylinders and number of lobes. After entering the data, additional options of inlet, exhaust and injector buttons with extract files and angle of opening for respective buttons appear. Then after, the distances between the lobes, bearings, etc., has to be modified. Later we can generate the model with new values.



**Fig-4:** GUI window design

### 3.2.2 Integration phase

In this phase of experimentation, the Creo model is integrated with GUI to achieve Design automation. The process of integration can be done using Creo API's (Application Programming Interface). Visual Studio is the API used in this study. It is the software that allows two applications to interface each other. The output variables from the design phase are the input variables of Visual studio. And the values in this frame work are the inputs to the CAD software.

Visual Basic API for Creo Parametric is an asynchronous application. For the application to communicate with Creo Parametric, you must set the PRO\_COMM\_MSG\_EXE

1. Click on Start > Settings > Control Panel.
2. Click on System. The System Properties windows open.
3. In the Advanced tab, click the Environment Variables button.
4. Add PRO\_COMM\_MSG\_EXE to System variables.

After successful integration with Visual Studio, the model gets updated when the values in the GUI are changed, which can be seen in the upcoming chapter.

## 4. RESULTS & DISCUSSIONS

As discussed in previous chapter, interfacing the Creo Model with VB API, the model is ready for automation. In order to change dimensions of the model, just enter the values in the GUI and Click Generate Model. The Model gets generated automatically with given values in the GUI.

After completion and debugging, the program was able to carry out as planned. By providing the necessary user inputs, the program is able to return the planned outputs. The end result of the project, an interactive software tool that assists the user

within the answer of the primary encountered problems when modelling a camshaft, was carried out as initially planned. It does simplify the task and saves the modelling time.

Step-by step model gets generated as per the input parameters in the user interface. The change in the model, before and after the model generation is shown in the fig.5.

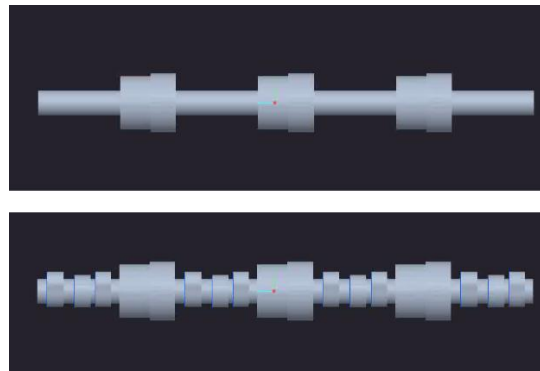


Fig-5: Before and after generation of model through GUI

In this project, we can clearly see the time difference in creating a model through conventional modeling and through programming language interfacing with CAD software. The modeling time difference can be shown through the flow chart as shown in the fig.6.

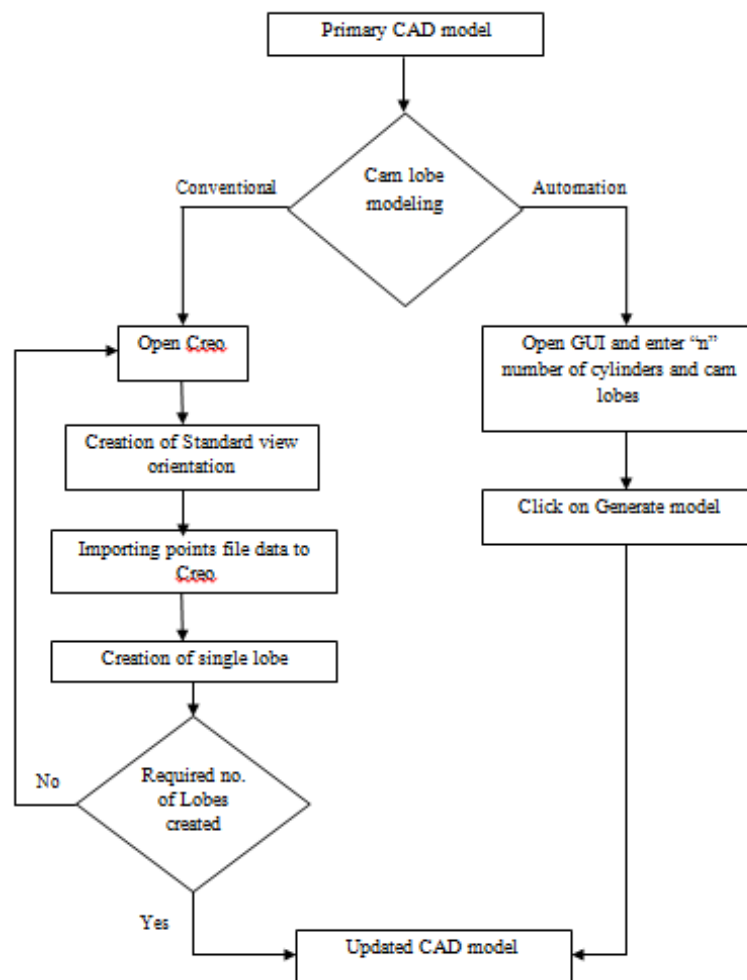
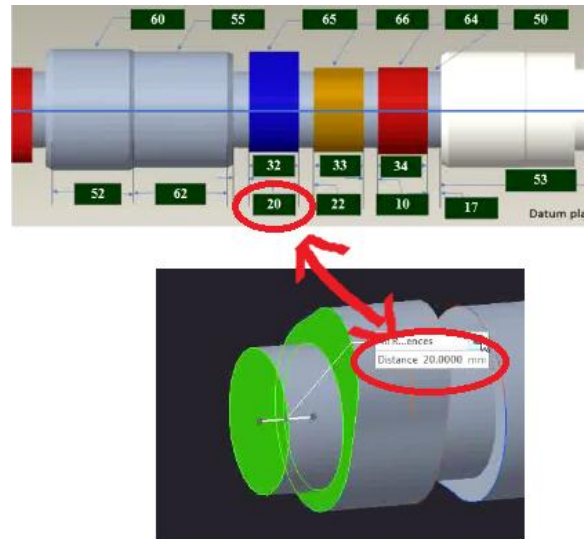


Fig-6: Flow chart of design process compared with conventional modeling and modeling using C#



From the flow chart we can easily observe the time taking through conventional modeling for each lobe, whereas, in GUI using programming language the time taking for modeling is too less. It can save the modeling time by more than 70% for the modeler. From the generated model, dimensions can be crosschecked with the model from given input as shown in fig. as one of the example, which gives the same value that we have entered in GUI.



**Fig-7:** Exact dimensions obtained from GUI to Creo model

## 5. CONCLUSIONS AND FUTURE SCOPE OF STUDY

According to the experimental study on design automation, following conclusions can be made

- Scope of programming languages in the mechanical design and modeling processes is for effective work in the industries.
- Use of programming languages helps in saving the time in design phase and decreases the time gap between designing and modeling.
- Parametric Modeling in CAD software useful for faster regeneration of models during the simulation with new dimensional values of a component.
- Integration of programming languages and CAD software helps the industries in this fast growing competitive world.
- Further, the integration of analysis phase in Automation, in future has a decent extension.

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